

Filip ROŽIĆ, Navalís d.o.o., Put Budeševa 5A, HR-21312 Podstrana, Croatia, filip.rozic@navalis.hr
Smiljko RUDAN, Fakultet strojarstva i brodogradnje, Ivana Lučića 5, HR-10000 Zagreb, Croatia, smiljko.rudan@fsb.hr

STRUCTURAL ANALYSIS OF GUIDE SHEAVE HOUSING ON BACKHOE DREDGER

Abstract

Backhoe dredgers are complex and distinguished ships. A 60m long, self-propelled Backhoe dredger in concern is used for excavation of soft soil in the Panama Canal. For lifting and positioning of the backhoe dredger, the pontoon is equipped with the three spuds: two fixed spuds are positioned close to the excavator, while the third spud, with a hydraulically-operated spud carrier, is positioned on the opposite site of the dredger. Spud lifting operation is performed by hydraulic winch and sheave housings. Guide sheave housing is an important part of the lifting operation and represents a non-typical, custom-made structural detail. Due to significant sheave forces and the importance of the guide sheave housing integrity, a structural analysis is performed to assess the overall strength, as well as the stress concentration in the critical hot-spots. A top-down FEM analysis is performed. Global FE model of the ship and two-level sub-models were generated and then hot-spot stress is being determined on two considered locations. IIW and DNV Rules have been used for stress extrapolation. The results were scattered and the consequences of these findings are commented in the article.

Key words: structural analysis, hot-spot, global model, sub-model, non-typical structural detail

STRUKTURNA ANALIZA KUĆIŠTA KOLOTURA NA JARUŽARU S GRABILICOM

Sažetak

Jaružari s grabilicom spadaju u kompleksne i istaknute brodove. Samopogonjeni, 60 metara dug jaružar s grabilicom, opisan u ovom članku, koristi se za iskopavanje mekog tla u Panamskom kanalu. Za podizanje, premještanje i pozicioniranje jaružara, trup broda opremljen je s tri nosiva stupa: dva fiksna stupa nalaze se u blizini grabilice, dok se treći, hidraulički pogonjeni stup, nalazi na suprotnoj strani jaružara. Podizanje stupa vrši se preko hidrauličkog vitla i kućišta kolotura. Kućište kolotura je bitan dio sustava podizanja stupa i predstavlja netipičan i nestandardan strukturni detalj. Zbog velikih sila u koloturama i važnosti koju ima kućište kolotura, provedena je numerička strukturna analiza u svrhu provjere čvrstoće kućišta, uključujući i proračun faktora koncentracije naprezanja u kritičnim detaljima kućišta. Analiza je provedena metodom konačnih elemenata po "top-down" principu. Generiran je globalni model broda te dvije razine podmodela, na temelju čega je određeno žarišno naprezanje za dva razmatrana detalja. Za ekstrapolaciju naprezanja korištena su IIW i DNV pravila. Raspršenost rezultata i utjecaj načina ekstrapolacije na dobivene rezultate komentirani su u članku.

Ključne riječi: strukturna analiza, žarišno naprezanje, globalni model, podmodel, netipični strukturni detalj

1. Introduction

Dredging is vital to the construction and maintenance of maritime infrastructure upon which economic prosperity and social well-being depends. Dredging industry has been developed locally in areas where maintenance of navigable waterways was needed (Netherlands, Denmark etc.) so it is not surprise that most of the dredging industry is situated in those areas. In recent years, dredging industry has been developed significantly, developing more sophisticated and specialized dredgers. One of these specialized dredgers, namely backhoe dredger, is described in this article. In addition to that, this article presents structural analysis of non-typical, custom made structural detail on backhoe dredger.

Backhoe dredger is a mechanical dredger that works by mechanically digging sediments from bottom surface through the use of a backhoe. Sometimes this type of the dredgers is equipped with a rock breaker and a TT pumps. Backhoe dredger is mostly used for moving a wide variety of materials, such as human waste, trash, gravel etc. They also help to keep the canals, harbors and marinas clean. Area of the application covers almost all soils from the soft silt to the sand or rocks.

Project of the backhoe dredger in concern was made by Dutch company IHC Merwede in cooperation with a Dutch shipyard NMC and Croatian company Navalís who did technical and CAM/CAD documentation. One of the tasks that were appointed to Navalís is to make a structural analysis of the guide sheave housing, a vital part of spud lifting operation.

Linear static FEM analysis has been performed and a result of that analysis has been presented in this article. In addition to the linear static analysis, hot spot stress has been determined for the two considered locations. For this analysis sub-modeling technique was used.

2. Description of the ship

The backhoe dredger “Alberto Alemán Zubieta” is of the assisted propelled type, capable of dredging by means of one pedestal mounted excavator, equipped with a 17 m boom, 12.0 m stick and 12 m³ bucket, Figure 1. Dredger is capable to deliver the soil either into barges moored alongside the vessel or directly on the sea-bed near the vessel, and of dredging sand, clay and blasted rock within the capabilities of the excavator. Dredger is capable of dredging in a range of 180°, swinging and dredging up to 90° to each side of centerline of the dredge at maximum dredging depth and reach. Vessel is equipped with three steel spuds; spuds are hoisted by means of hydraulic winches. The forward spud winch is mounted on the spud carrier. Aft spud winches are mounted in the winch rooms. The spuds can be used for anchoring the vessel during dredging and laying idle, lifting the pontoon to reduce the pontoon’s draught in order to increase the anchoring force and to reduce the influence of waves and currents on the dredger. Spuds can also be used for movement of the pontoon by means of the spud carrier. In working condition the vessel can be partly lifted to reduce the pontoon's draught in order to increase the anchoring force, and to reduce influence of waves and current on the dredger. Pontoon moves by means of a spud carrier. Spud carrier moves by means of a hydraulic winch. The deck shape near the excavator is round to maintain a dredging area of approx. 180 degrees. To achieve a sufficient bow height the fore deck is raised. Barge mooring winches are arranged on the main deck for the barge shifting during dredging. Two hydraulic cranes are provided on main deck to facilitate repairs and handling of the heavy weights, including bucket changing. Two generator sets are provided supplying the electric power to the auxiliary consumers. For the emergency and harbor operation an emergency/harbor generator set is provided. Hydraulic power for the pontoon mounted hydraulic equipment is supplied by the excavator crane. One hydraulic backup power pack is provided in the engine room for the pontoon functions for when the excavator is not in use. In case of the engine failure of the excavator engine, the backup power pack can be connected to the excavator to restore the excavator to its rest position. One wheelhouse is arranged. Wheelhouse is provided with propulsion and spud movement controls. The vessel is equipped with one “Man Over Board” boat. In the fore and aft

ship trim tanks are arranged in order to reduce the vessel's trim to even keel when in working position.



Fig. 1. IHC Backhoe dredger “Alberto Alemán Zubieta”

Main particulars of the backhoe dredger “Alberto Alemán Zubieta” are shown in Table 1.

Table 10. Backhoe dredger Main particulars

Length overall pontoon	60.20 m
Length between perpendiculars	58.25 m
Breadth, moulded	23.0 m
Depth, moulded	4.0 m
Design draught hull	3.0 m
Maximum jack up height	1.0 m
Deadweight "all told" at 3.00 m draught	723.5 tons
Maximum depth reach on a draught of 3.00 m	19.5 m
1 dredge diesel engine, 1800 rpm	2000 kW
2 AC auxiliary generators, output of each	285 kVA
1 AC emergency/ harbor generator	100 kVA
Complement	28 persons
Tank storage capacity of diesel oil app.	275 m ³

3. Guide sheave housing

Lifting the pontoon allows the higher digging force to be applied and reduces the impact of waves and current on the hull of the dredge. By lifting the dredger, a stable work platform for the excavator is created. Lifting is achieved by hoisting wires running over sheaves and operated by spud hoisting winches operated by hydraulic motors. Spud lifting system is capable of lifting the dredger up to 1.0 m out of its floating position. Guide sheave housing is one part of that spud lifting system and represents vital part of lifting operation. The wires from the spud winches run via sheaves, through main deck, to sheaves near the spuds. From these sheaves the wires run vertically via the top and bottom sheaves of the spuds to sheaves on the other side of the spuds. Via these sheaves and the top and bottom sheaves of the spuds the wires run to the end wire connections, see Figure 2. Wires are roved in such a way that the same winch is used for lifting and lowering the spuds and lifting the vessel. Spud lifting/lowering speed is approximately 4.0 m/min. For each spud (6), system consist of a hydraulic winch (1), with double wire (one for hoisting and one for the spud lowering); fixed guide sheaves (2) running the wires to top and bottom of the spuds, two sheaves on top (3) and two sheaves in bottom (4) of each spud; one turning sheave (5) on the guide frame; a

tensioning and fastening system, fitted in the dead part of the spud hoisting wire. See Figure 2 for visual representation of the above mentioned.

At the bottom, the spuds are provided with a heavily constructed steel peak. The steel peak allows penetration in hard soils and has sufficient strength to withstand the maximum spud load. The spuds are provided with a locking system for locking the spuds in maximum hoisted position. All three spuds are identical. Spuds are provided at top and bottom with sheaves for hoisting and lowering. Guide sheave housing has a guide mechanism to keep cable within the sheave groove in the event of a slack cable condition.

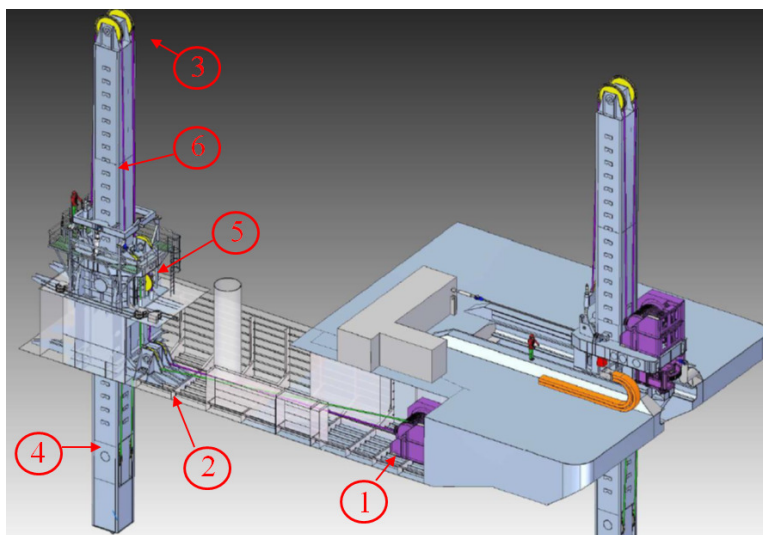


Fig. 2. Isometric view on spud lifting system

The spud winch has a following specification: maximum nominal line pull 1 250 kN at 16 m/min, maximum holding force 2 000 kN, drum diameter approx. 1910 mm, drum width approx. 2030 mm, steel wire diameter 80 mm, steel wire type 6 x36 WS + steel core. 1.960 N/mm², breaking strength approx. 4470 kN.

Subject of this paper is one part of this system i.e. guide sheave housing. Figure 3 shows the model of a guide sheave housing (left figure) with the cross section view of the sheaves (right figure). Two sheaves are mounted per housing. Two guide sheave housing exist, situated on portside and starboard side of ship, mirrored. Housing plate thickness varies and is 8, 10, 12, 15, 20, 25, 30, 40 and 60 mm. Sheave material is a cast steel and the housing plate material is a high tensile steel S275JO, S355JG.

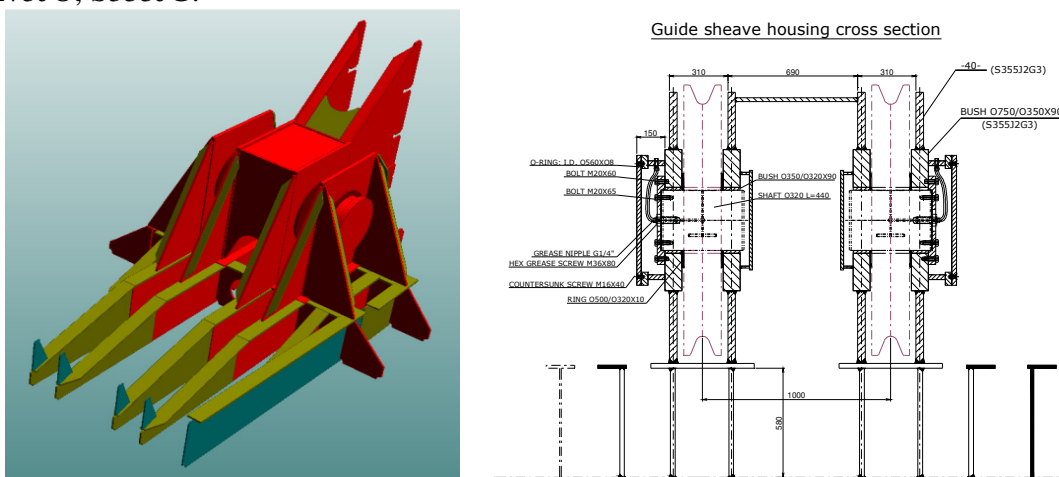


Fig. 3. Guide sheave housing (left) and cross section (right)

4. Finite element analysis

Finite element analysis is performed using Femap/Nastran software in three stages, using top-down approach:

1. Global model analysis,
2. First level sub-model analysis,
3. Second level sub-model analysis.

Global model analysis is performed to verify stress level in the hull girder and primary supporting members, including deflections and to check buckling capability of primary supporting members. Buckling analysis is not presented in this article. Global finite element analysis allows detailed investigation of the structure response at any location, thereby providing assurance that potential problem areas are identified at the earliest possible stage.

First level sub-model represents a part of global model where guide sheave housing is situated. First level sub-model has been generated with finer element mesh size to capture stress flow in that area. Sub-modelling technique has been used for this model and prescribed displacements from global model have been applied on model boundaries.

Second level sub-model represent model for determining hot spot stresses used in fatigue calculation. Model had been generated with very fine mesh defined by IIW [1] and DNV [2] rules. For this model sub-modelling technique has also been used and displacements from first level sub-model have been applied on model boundary. Next chapters describe these three stages, in details.

4.1 Global model

The purpose of the global model is to obtain global deformation and stress distribution. Backhoe dredger global model has been generated with coarse finite element mesh (Figure 4) and it is used to represent global behaviour of primary structure. Mesh size depends on the shape of the structure, stiffener distance, size of the stiffeners, a position and dimensions of the openings and other characteristics of the primary construction elements. Approach of determining global model is given by Bureau Veritas Rules [3]. Average finite elements size is 500x500 mm but on the openings mesh size is smaller. Element size for profiles is equal to profile height. Model has 24185 elements and it consists of triangle, rectangle and beam elements where 85% of elements are rectangular. Global model dimensions are Length x Width x Height = 54550 x 22920 x 5100 mm.

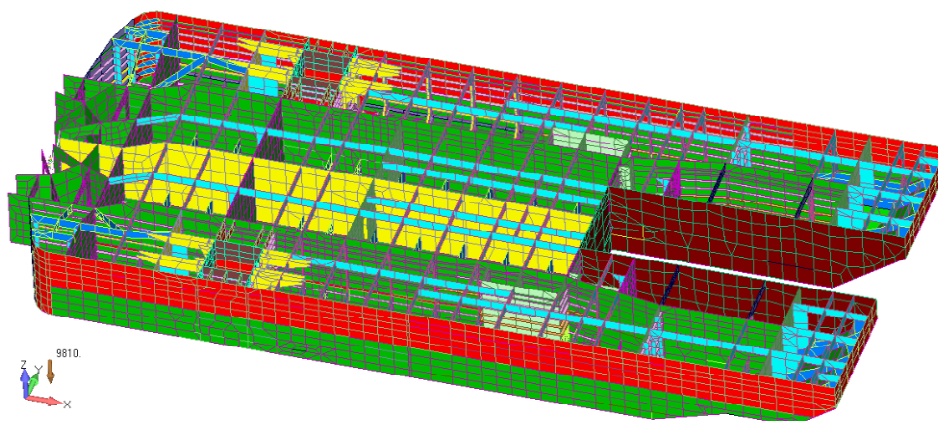


Fig. 4. Global dredger model (main deck hidden)

Global model includes all important elements of structure; longitudinal and transverse bulkheads, side and longitudinal girders on deck, frames, brackets, shell plating, bottom plating, main deck, tanks, profiles, stiffeners etc. Brackets, stiffeners, different foundations, openings that are not important for analysis have been excluded from model.

4.2 Global model analysis

Excavator, spud upper construction, spuds, spud carrier, living quarters, fender, equipment have not been modelled. Their weight is replaced with the appropriate load. On the global model following loads have been applied:

- Hydrostatic pressure,
- Living quarter section (260 kN),
- Excavator load (5800 kN),
- Upper spud support structure (195 kN),
- Funnel construction (195 kN),
- Fender and equipment (267 kN),
- Dredger self-weight.

Boundary conditions are assumed so that the model is fixed on a spud position. For the two fixed spuds opening holes have been fixed and for movable spud, opening around spud carrier has been fixed, in a position when spud is at end position (unfavourable case), Figure 5.

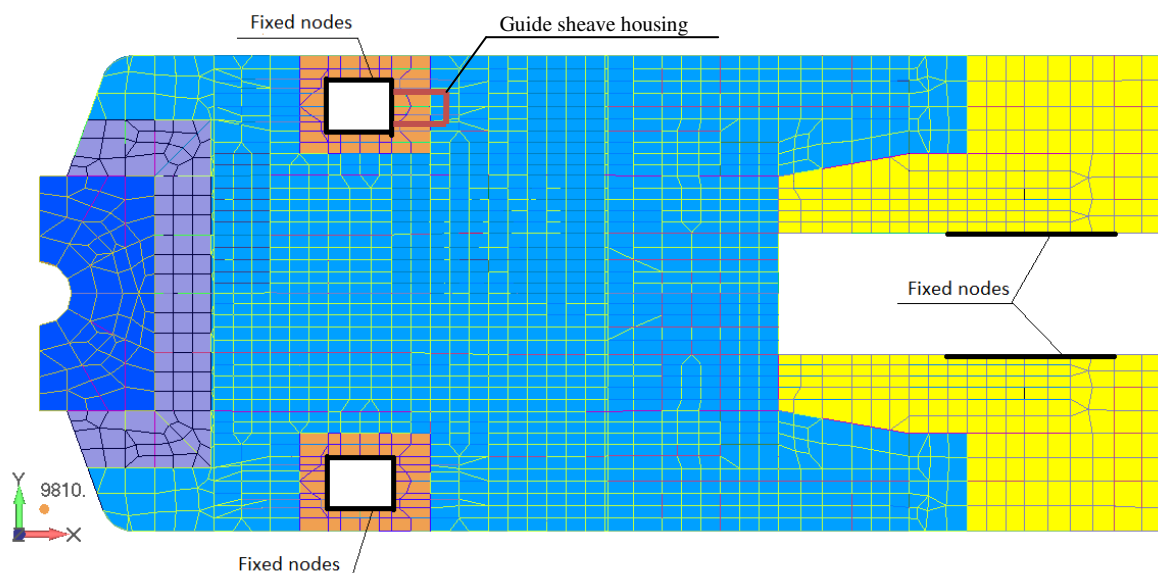


Fig. 5. Boundary conditions.

Position of the guide sheave housing on a global model is shown by red square on Figure 5. It is situated beneath main deck. Allowable stress has been determined according to “Common Structural Rules” [4] where allowable stress for coarse mesh is determined by material factor k :

$$\sigma_{Al} = \frac{235}{k}$$

Table 11. Allowable stress

Material designation	Yield point R_{eH} (MPa)	Material factor k	Allowable stress for “coarse mesh” (MPa)
GR-A	235	1	235
S275JO	275	0,89	264
S355JG	355	0,72	326,4

4.3 First level sub-model

In a second phase of the analysis, a ship section where the sheave housing is situated has been modeled by the fine mesh of finite elements in order to obtain more accurate stress distribution,

Figure 6. Main purpose is to analyse structural behavior of the local structure surrounding the guide sheave housing. Displacements of the global model, i.e. coarse mesh model, are used as boundary condition. For the first level submodel fine mesh model is thus forced into the same displacement pattern as the coarse model corresponding to ship global deformation. In addition to that, local hydrostatic pressure load is included in the local model, producing local stress. Local hydrostatic pressure load is acting on hull plating. Mesh size of the model is 100x100 mm element size, but in the area of the guide sheave housing, mesh size is even smaller. Model dimensions are Length x Width x Height = 10400 x 5700 x 4520 mm. Model has 51137 elements. Detail of guide sheave housing model is shown on Figure 6 right.

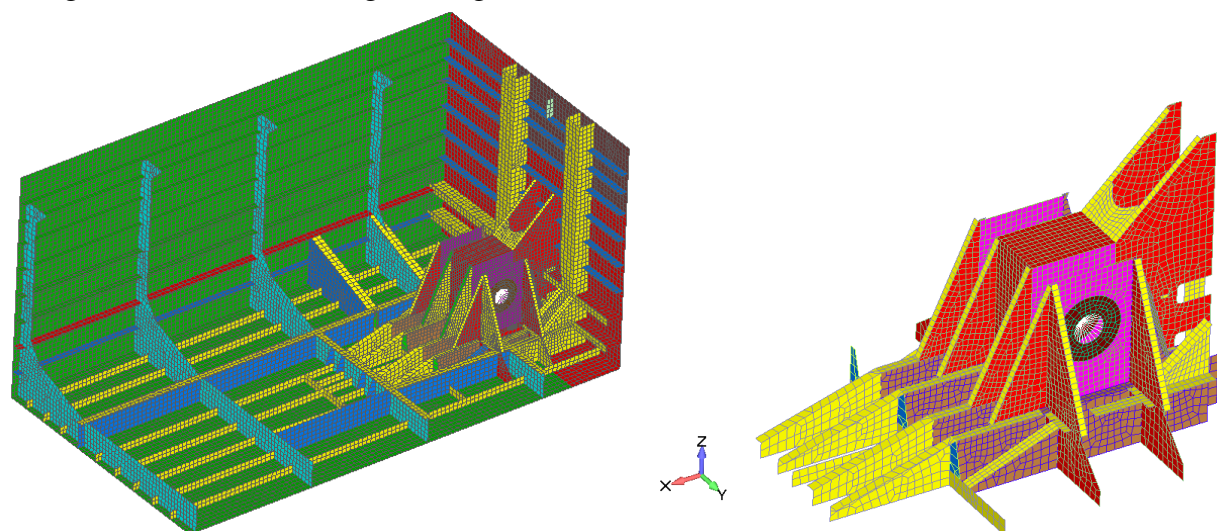


Fig. 6. Guide sheave housing - first level sub-model (main deck and hull hidden)

4.4 First level sub-model analysis

Load on the guide sheave housing is determined as maximal force acting in the steel wires and that force is 2100 kN where total resultant force in housing is 2970 kN. Total force is acting on both sheaves simultaneously. Both sheaves and steel wires are assumed to be undeformed. Load applied on the model is a sheave force of 2970 kN, local pressure on hull plating and global displacement acting on edge of model. The allowable stress has been determined according to “Common Structural Rules” [4], where allowable stress for the fine mesh is determined by material factor k :

$$\sigma_{Al} = \frac{280}{k}$$

Table 12. Allowable stress

Material designation	Yield point R_{eH} (MPa)	Material factor k	Allowable stress for “fine mesh” (MPa)
GR-A	235	1	280
S275JO	275	0,89	314,6
S355JG	355	0,72	388,8

Linear static FEM analysis show that the highest von Mises stress in the guide sheave housing is 131.5 MPa under prescribed load, figure 7. Maximal total translation is 1.1 mm. Since the housing plates are made from the high tensile steel it can be concluded that highest stress is under allowable limit with a safety factor of 2.1.

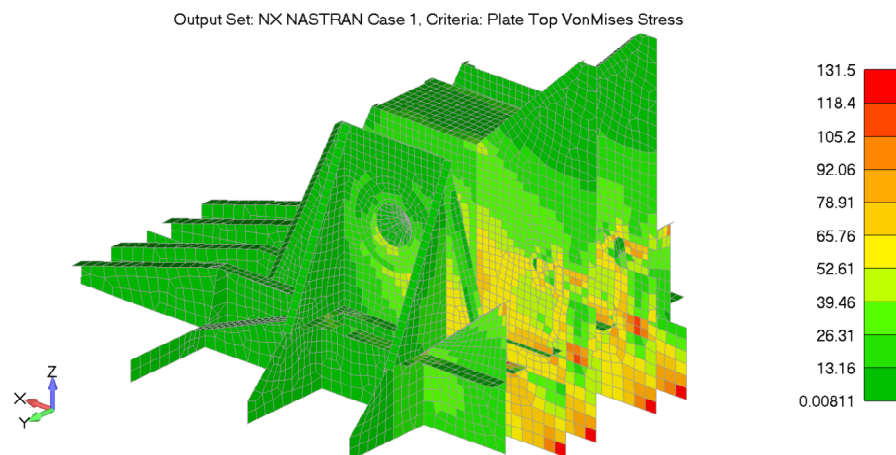


Fig. 7. Von Mises stress, MPa

4.5 Second level analysis – hot spot stress approach

Second level sub-model is used to determine the hot spot stress in critical structural details. Very fine mesh is needed to determine hot spot stresses in the areas of a geometric discontinuity where stress concentration is expected to occur.

The structural hot spot stress method is still a relatively new method. It has not been used as wide as nominal stress method. It is suited to use in the following situations:

1. When fatigue design of a new welded detail or fatigue assessment of an existing detail is required, but the detail is not a standard detail that can be found in the tables provided by the design code.
2. In complex details where a clear definition for the nominal stress cannot be made
3. When modelling a component with a detailed finite element model, calculation of the nominal stress is not trivial and at least needs some post-processing.

For determination of the hot spot stress it is necessary that the element mesh is adequately small to more accurately describe stress distribution. Fine element mesh is very sensitive on the choice of size and type of the finite element. With a lowering of the element size stress could unrealistically increase, thus elements size has to be predetermined and the hot spot stress is to be determined by the means of extrapolation. IIW and DNV give the guidelines on the element size and a way to perform extrapolation. Hot spot stress approach is only applicable to the situations where the potential mode of failure is by fatigue crack growth from the toe of a weld. In general three types of weld toe failure can be identified, Figure 8:

1. Type a) used for welds at the end of longitudinal attachment (weld toe or end on loaded plate surface)
2. Type b) used for welds on or around a plate edge (weld toe on plate edge),
3. Type c) used for welds transverse to loading (weld toe on loaded plate)

In two cases, (a) and (c), it is generally accepted that the stress distribution approaching the weld toe depends, amongst other things, on the plate thickness. For this case the structural hot spot stress is determined using reference points and extrapolation equations. Element size can either be $0.4 t \times t$ or $t \times t$, where t is plate thickness. For the element size $0.4 t \times t$ extrapolation is performed by evaluation of nodal stresses at two reference points $0.4 t$ and $1.0 t$ and linear extrapolation to the hot spot or by quadratic extrapolation at the three reference points $0.4 t$, $0.9 t$ and $1.4 t$. Last method is recommended in a case with pronounced non-linear structural stress increase toward the hot spot.

In the case of type (b) the stress distribution is not dependent on the plate thickness so reference points are given at the absolute distance from the weld toe or from the weld end if the weld does not continue around the end of the attached plate. Element size is 4 x 4 mm or 10 x 10 mm; in case of element size 4 x 4 mm extrapolation is performed at three reference points at 4 mm, 8 mm and 12 mm and quadratic extrapolation is used. In case of 10 x 10 mm element size evaluation of the stresses is done at the mid-side points of the first two elements and linear extrapolation is used.

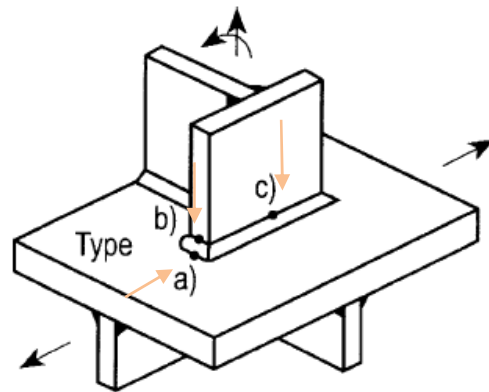


Fig. 8. Types of hot spots in the welded structures

Linear static FEM analysis is used to determine the structural hot spot stress. The mechanical properties of the material are selected to be the same as the parent metal. Both shell and solid element models are generated. The elements type and mesh are modelled in a way to capture the formation of abrupt stress gradient at the weld toe. But at the same time only the linear stress distribution in the plate thickness needs to be determined. Rules and guidelines on modelling of structure have been determined by IIW recommendations.

In a shell element model, 4-node or 8-node isoparametric elements are commonly used. Welds are usually not introduced into the model but they can be. Welds in shell model can be modelled by increasing thickness of shell elements in weld region to compensate increased thickness in that region or by modelling weld with shell elements. One other way is by creating rigid links between corresponding nodes on two joined plates. Solid elements are also used for determining hot spot stress and are recommended in modelling complex structures. Usually prismatic solid elements with quadratic shape function and reduced integration are used. Another less selected choice is tetrahedral elements. These elements are desirable because of easy and automatic mesh generation technique. In this case, a sufficiently fine mesh of second order tetrahedral elements should be used. When using solid elements, modelling of the weld is always recommended.

In general, chosen element type should allow for a linear stress distribution through the thickness. This can be achieved by 8-node shell elements or 20-node solid elements with reduced integration. A single layer of solid elements through the plate thickness is sufficient. The size of the elements in the vicinity of the hot spot should be chosen such that the extrapolation points coincide with the elements integration points or nodal points. Largest allowed aspect ratio for elements is limited to 3. Transition from fine mesh at the weld region into coarser mesh in other parts of the model should be gradual.

4.6 Second level sub-models

Second level sub-models have been modelled with a very fine finite element mesh. On a model boundary displacements from the first level sub-model have been applied. Hot spot stress extrapolation was performed for two structural details. First structural detail, detail A represents junction of the bracket and the lower face plate while detail B represents a plate with 35 mm radius.

For each detail, two types of finite element mesh have been generated; shell and solid. For model with the solid elements, welds have also been modelled. Size of the elements is determined according to recommendations of IIW [1] and DNV [2].

4.6.1 Detail A analysis

Two finite element models of the Detail A have been made, 8-node shell and 20-node solid, see Figure 9. For the detail A hot spot stress is determined by a linear extrapolation from the middle nodes of the first two finite element away from the hot spot. Way of the extrapolation matches type b) method. Mesh size is 10x10 mm for the shell finite element model and 10x10x10 mm for the solid finite element model. Extrapolation points are situated on the middle nodes 5 and 15 mm away from the hot spot. For both details direction of the extrapolation is from top to bottom.

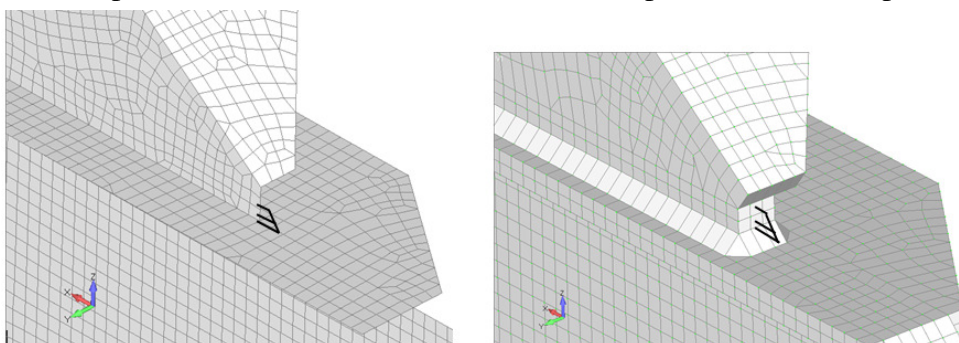


Fig. 9. Detail A, shell and solid mesh

Hot spot stress is determined by equation:

$$\sigma_{HS} = 1,5 \cdot \sigma_{5 \text{ mm}} - 0,5 \cdot \sigma_{10 \text{ mm}}.$$

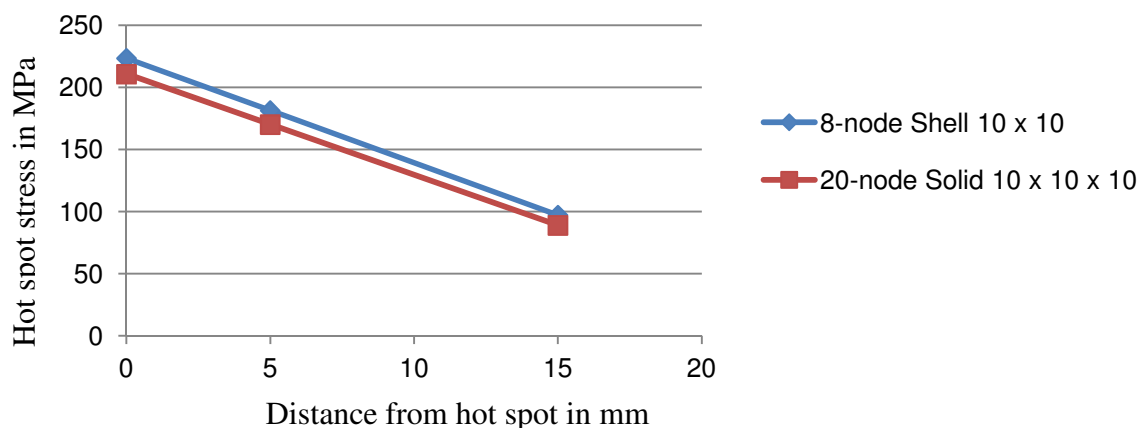


Fig. 10. Hot spot extrapolation for detail A

Figure 10 represents a hot spot stress extrapolation for Detail A. As noticed, solid elements with the elements size of 10 x 10 x 10 mm show smaller values of the extrapolated hot spot stress then sub-model modelled with the shell elements of size 10 x 10 mm. Reason for that is additional weld stiffness included in 3D sub-model. Extrapolated stress for the shell sub-model is 229 MPa and for the solid sub-model is 210 MPa.

4.6.2 Detail B analysis

For detail B, Figure 11, way of the extrapolation matches type c) method. Mesh size is $0,4t \times t$ where t is plate thickness. For solid element mesh, mesh size is $0,4t \times t \times t$. Hot spot stress extrapolation is performed by linear extrapolation from two nodes at $0,4t$ and $1t$ distance from hot

spot. Finite elements used are second order elements with element aspect ratio no more then 1:3. Also mesh size $t \times t$ is applicable where hot spot stress is determined by linear extrapolation between nodes at $0,5t$ and $1,5t$ distance from hot spot. Three models have been made, one 20-node solid model with finite element size $10 \times 25 \times 25$ mm and two 8-node shell models with finite element size 25×25 mm and 10×25 mm.

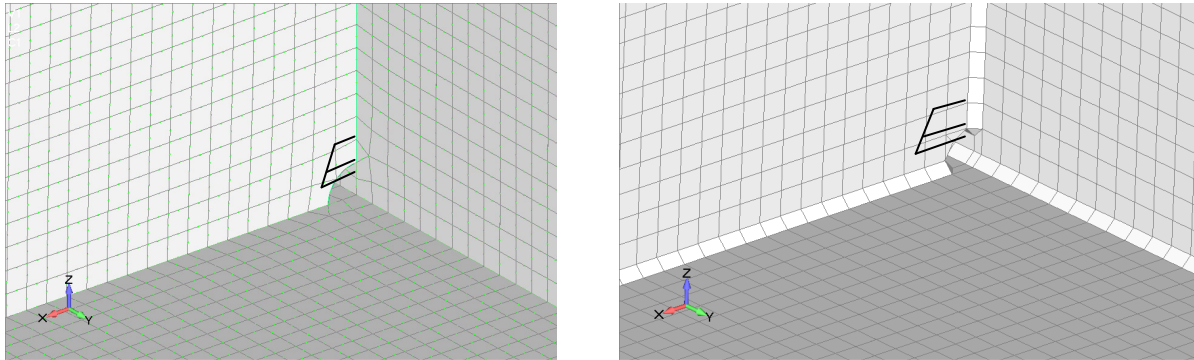


Fig. 11. Detail B, shell and solid mesh

For $0,4t \times t$ mesh size, hot spot stress is determined by equation:

$$\sigma_{HS} = 1,67 \cdot \sigma_{0,4t} - 0,67 \cdot \sigma_{1t}.$$

And for $t \times t$ mesh size by:

$$\sigma_{HS} = 1,5 \cdot \sigma_{0,5t} - 0,5 \cdot \sigma_{1,5t}.$$

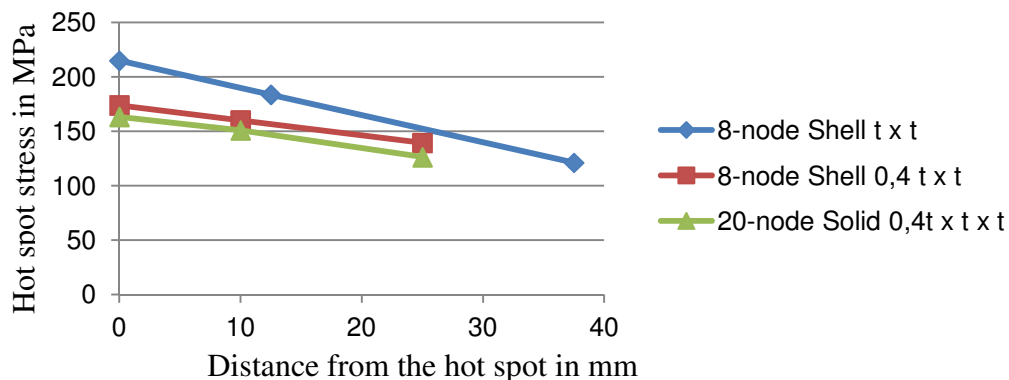


Fig. 12 Hot spot extrapolation for detail B

Figure 12 represents a hot spot stress extrapolation for Detail B. Similar to Detail A, solid elements with elements size of a $0,4t \times t \times t$ mm show smaller values of extrapolated hot spot stress then sub-model modelled with the shell elements. Reason is the same as for Detail A - additional weld stiffness included in 3D sub-model. Two additional shell sub-models have been made with the different mesh size and the hot spot stress extrapolation method. As shown in figure, results deviate from each other, meaning that extrapolated stress for element size of the $t \times t$ show higher hot spot stress value then the sub-model modelled with the shell elements of size $0,4t \times t$. From this it can be concluded that uncertainties in computed structural stress exists mainly due to element properties and sizes.

5. Conclusion

This paper describes structural analysis of one non-standard structural detail on Backhoe dredger. Three-stage analysis was performed using top-down approach; global model analysis, first level sub-model and second level sub-model analysis. Global model analysis was performed to

determine dredger global behaviour and to determine the global deformation that was then applied on the first level sub-model representing guide sheave housing. Second level model was made for the purpose of determining the hot spot stress. On the second level sub-model a deformation of the first level sub-model has been applied. The aim of the paper is to investigate the application of hot spot stress approach on two structural guide sheave housing details. Two details were investigated as shown in previous chapters and cumulative result can be found in table 4.

Table 13. Hot spot stress results

Detail	Element	Extrapol.	Element size	Extrapolation points	Hot spot stress (MPa)
A	Shell	Linear	10 x 10 mm	Midside node at 5 and 15 mm	229,6
A	Solid	Linear	10 x 10 x 10 mm	Midside node at 5 and 15 mm	210,6
B	Solid	Linear	10 x 25 x 25 (0,4t x t x t)	Midside node at 12,5 and 25 mm	171,56
B	Shell	Linear	25 x 25 mm (t x t)	Midside node at 0,5t and 1,5t	222,75
B	Shell	Linear	10 x 25 mm (0,4t x 1t)	Node at 0,4 t and 1t	181,3

The structural hot spot stress approach is a relatively new approach for fatigue assessment of welds. The method is advantageous compared to the traditional nominal stress method mainly because of its ability to assess more types and variations of the structural details. It incorporates the effect of structural geometry into the local stress ranges at the welds and predicts the fatigue life based on these local stress ranges. The structural hot spot stress method has been accepted widely in the shipbuilding and offshore industry for many years as an efficient and reliable method to assess the fatigue strength of welded steel detail. It is expected that this method leads to more realistic fatigue life assessment of welded details. Although many advantages of this method exist it should be pointed out that scatter and non-consistency of results are present and addition research and scientific effort should be made which would allow more reliable results and guidelines. Noticeable difference in computed structural hot spot stress between the solid element models and shell element models was found. Shell element models predict higher hot spot stress level leading to shorter fatigue life then solid element models do. This is on the safe side of calculation. Uncertainties in the stress evaluation are also found mainly due to the element properties and size. Till better guidelines are defined, analyst can rely on recommendations of classification society's or scientific research on the standard existing details.

Disclaimer

The drawings, figures and opinions incorporating the original design and conceptions of the author is in property of NAVALIS d.o.o. and should not be disclosed, copied, duplicated or in any way made use of except with approval of NAVALIS d.o.o.

References

- [1] ...: "Recommendations for fatigue design of welded joints and components", International Institution of Welding, October 2008.
- [2] ...: "Fatigue Assessment of Ship Structures" Det Norske Veritas, October 2008.
- [3] ...: "Rules for the Classification of Steel Ships", Bureau Veritas, France, 2009.
- [4] ...: "Common structural Rules for Bulk carriers", Rules, January 2006.

Albert ZAMARIN, Faculty of Engineering University of Rijeka, Vukovarska 58, HR-51000 Rijeka, Hrvatska, zamarin@riteh.hr

Tin MATULJA, Faculty of Engineering University of Rijeka, Vukovarska 58, HR-51000 Rijeka, Hrvatska, tin.matulja@riteh.hr

Nikša FAFANDJEL, Faculty of Engineering University of Rijeka, Vukovarska 58, HR-51000 Rijeka, Hrvatska, niksaf@riteh.hr

Darko SRZENTIĆ, Brodoplan d.o.o., Kumičićeva 13, HR-51000 Rijeka, Hrvatska, darko.srzentic@gmail.com

PROCJENA ČVRSTOĆE BALASTNE KOBILICE REGATNE JEDRILICE

Sažetak

Kobilica jedrilice je izdanak trupa koji ima presudnu važnost u postizanju performansi jedrilice. Uloga kobilice je osiguravanje stabiliteta, održavanje kursnog kuta i brzine, pri čemu presudnu ulogu imaju oblik i težina kobilice. Moderne jedrilice imaju kobilice u obliku peraje, koje se često koriste kod jedrilica za krstarenje, budući daju dobre performanse, a relativno su jeftine i jednostavne izvedbe. Kobilice na regatnim jedrilicama su duže i vitkije u svrhu postizanja znatno boljih performansi, zbog čega je pri odabiru oblika i mase kobilice potrebno voditi računa o dimenzijama unutarnje strukture koja mora osiguravati dovoljnu krutost i čvrstoću. U radu je prikazan projektni pristup odabiru i proračunu parametara lista i bulba kobilice, kao što su geometrija, masa i položaj težišta. Za dva predložena modela (L-bulb i T-bulb) proveden je proračun čvrstoće prema pravilima ISO-12215. Posebna pažnja je posvećena spoju kobilice i trupa, tj. odabiru vijaka. Za oba modela izvešena je analiza globalne čvrstoće struka kobilice primjenom metode konačnih elemenata, a dobiveni rezultati pokazuju deformacije i naprezanja u granicama dozvoljenih.

Ključne riječi: regatna jedrilica, balastna kobilica, strukturna analiza MKE

BALAST KEEL STRENGTH ASSESMENT OF A RACING SAILING YACHT

Abstract

Ballast keel is a hull appendage which is crucial in achieving the sailing yachts performance. The main function of the keel is to ensure stability, maintaining the ship's angle and speed, with the most important characteristics the shape and weight of the keel. Modern sailboats have keel-shaped fins, which are often used on cruisers as provide good performance and are relatively cheap and simple in design. In order to achieve the best or optimal performance, racing sailing yachts are designed and constructed with longer and slender keels with bulb. Therefore, during design process special attention should be paid to shape and weight selection as well as to provide required stiffens and strength. Within the paper, design approach is presented on keel and bulb parameters selection and calculations such as geometry, mass and center of gravity. For the two proposed structural models, first with L-bulb and second one with T-bulb, strength calculation is presented according to ISO-12215 rules. Special attention is paid to keel to hull connection, i.e. screws dimension calculation and selection. Finite elements global strength analyses of the keel internal structure is performed for the both of the structural models. Presented results, in form of deformations and stresses are discussed and are within allowable limits.

Key words: racing sailing yacht, ballast keel, FEA